

NASA Technical Memorandum 102480
AIAA-90-0488

A Numerical Simulation of the Flow in the Diffuser of the NASA Lewis Icing Research Tunnel

Harold E. Addy, Jr.
Lewis Research Center
Cleveland, Ohio

and

Theo G. Keith, Jr.
University of Toledo
Toledo, Ohio

Prepared for the
28th Aerospace Sciences Meeting
sponsored by the American Institute of Aeronautics and Astronautics
Reno, Nevada, January 8-11, 1990



(NASA-TM-102480) A NUMERICAL SIMULATION OF
THE FLOW IN THE DIFFUSER OF THE NASA LEWIS
ICING RESEARCH TUNNEL (NASA) 11 p CSCL 14B

N90-15965

Unclas
0261715

G3/09



A NUMERICAL SIMULATION OF THE FLOW IN THE DIFFUSER OF THE NASA LEWIS ICING RESEARCH TUNNEL

Harold E. Addy, Jr.*

NASA Lewis Research Center, Cleveland, Ohio

Theo G. Keith, Jr.**

University of Toledo, Toledo, Ohio

In this paper, the flow in the diffuser section of the Icing Research Tunnel at the NASA Lewis Research Center is numerically investigated. To accomplish this, an existing computer code is utilized. The code, known as PARC3D, is based on the Beam-Warming algorithm applied to the strong conservation law form of the complete Navier-Stokes equations. The first portion of the paper consists of a brief description of the diffuser and its current flow characteristics. A brief discussion of the code work follows. Predicted velocity patterns are then compared with the measured values.

INTRODUCTION

The diffuser section of the Icing Research Tunnel (IRT) at the NASA Lewis Research Center has been used in the past to perform a limited amount of testing. However, measurements have shown that the flow quality is highly nonuniform and is unsuitable for many aerodynamic tests. This poor flow quality probably adversely affects the overall performance of the IRT although the airflow in the test section appears to be well behaved. Because of this it was decided that there was a need to better understand the peculiar flow characteristics in the diffuser of the IRT. This understanding may suggest methods which may be used to improve the flow quality. If this objective is realized, the diffuser can be more widely used as a second, larger, but lower speed, test section. Also, the study may help to improve the overall performance of the tunnel.

Diffusers are used to decelerate flows such that static pressure can be recovered and, in the case of wind tunnels, friction losses can be reduced. A number of studies have been performed to gather

information on performance and characteristics of diffusers. However, much of the fundamental data in the literature is from axisymmetric and/or two-dimensional diffuser studies. Little information exists for fully three-dimensional diffusers. The IRT diffuser, having four flat walls which all diverge at constant angles, is of this three-dimensional type.

The study discussed herein is a continuation of the work reported in Ref. 1. That paper details the flow measurements which were made in the IRT to establish quantitatively some of the flow characteristics of the tunnel and to form a basis of comparison for computational studies of the flow. The initial computational results which precipitated the current numerical simulation are also discussed.

Numerical simulation techniques are increasingly being used to model and study internal flows. Two such techniques have been used to model the flow in the diffuser of the IRT. The first technique, which has been incorporated into a computer code known as PEPSIG, uses a forward marching procedure to solve a parabolized form of the Navier-Stokes equations. This procedure has the capability of solving three-dimensional, turbulent, subsonic flow problems. The PEPSIG code has had

* Aerospace Engineer, *Icing and Cryogenics Technology Branch*, Member AIAA

**Professor, *Mechanical Engineering Dept.*
Associate Fellow AIAA

success in the past in accurately modeling internal flows in inlets and ducts. However, it has some restrictions which limit its usefulness when applied to the IRT geometry. One limitation concerns its inability to model the rectangular shaped cross section of the IRT. Instead it used a superelliptical cross section which rounds each corner. This is similar to adding fillets to the cross section of the tunnel. Another limitation concerns its limited grid size, particularly in the axial direction. This restricts the amount of the test section and diffuser which could be accurately modelled. The flow predicted by the PEPSIG code was markedly different from that which was measured in the tunnel. The predicted velocity profiles in the diffuser were much flatter and more uniform than those measured. Also, the code computed a much smaller flow speed in the four corners of the tunnel than the measured values. PEPSIG predicted that some separation was occurring in the corners whereas the measurements strongly suggest that separation, if it does occur, would not occur in the corners, as will be discussed later.

Due to the lack of agreement between the computed results from PEPSIG and the measured values, a second numerical simulation was employed. Application and discussion of this second computer program, known as the PARC3D code, is the subject of this paper.

IRT GEOMETRY AND MEASURED FLOW CHARACTERISTICS

A diagram of the IRT is shown in Fig. 1. It is a closed loop wind tunnel currently capable of operating at test section airspeeds of up to 300 mph. A large refrigeration system allows the tunnel to operate at total temperatures as low as minus 20°F. Air flows from a settling chamber through a nozzle with a contraction ratio of 14.13:1.0 into the test section which is 20 feet long and has a rectangular cross section that is 6 feet high by 9 feet wide. The settling chamber and contraction section are also rectangular in shape.

The portion of the IRT which forms the diffuser is denoted in Fig. 1. This portion is 81.5 feet long and consists of four flat, straight walls each of which diverges from the centerline at an angle of 2.5°. The diffuser entrance is 6 feet high and 9 feet wide while the exit is 13.469 feet high and 16.469 feet wide. The expansion ratio of the diffuser is 4.11:1.0.

As mentioned earlier, the flow in the test section and the diffuser entrance appears to be well behaved. Figure 2 shows measured vertical velocity profiles at the centerline of the diffuser entrance for test section speeds of 150 and 300 mph. Figure 3 shows measured vertical velocity profiles in five equally spaced locations across the diffuser entrance for a test section speed of 200 mph. In each of these figures the vertical location of the local measure-

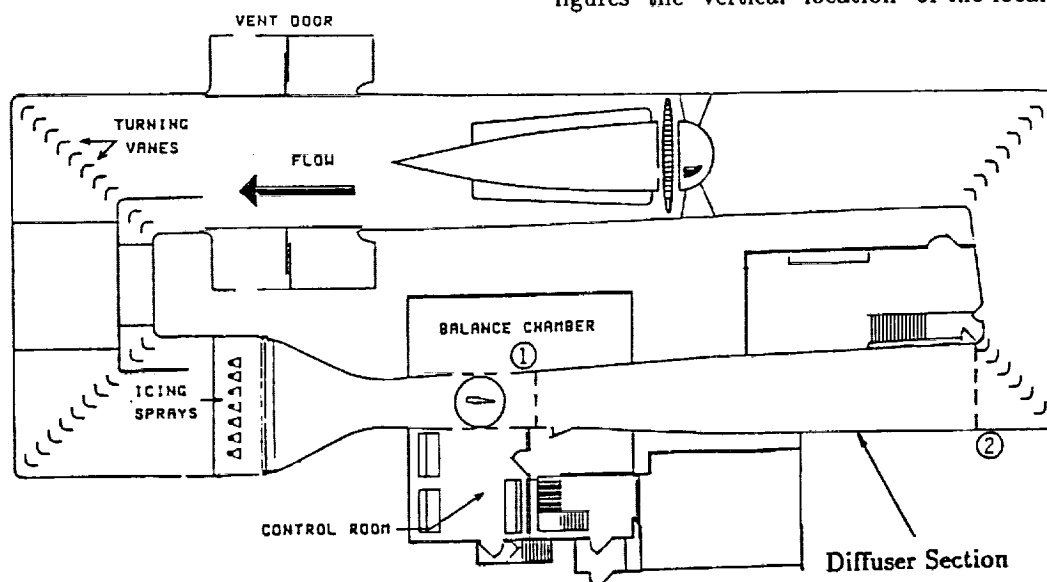


Fig. 1 Icing Research Wind Tunnel (① = diffuser entrance, ② = diffuser exit)

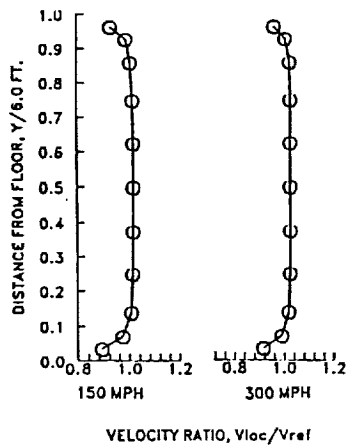


Fig. 2 Measured Profiles at Diffuser Entrance

ment above the floor of the tunnel is plotted against the local value of the measurement which is normalized by the tunnel reference value as measured by the facility Pitot-static tube located in the test section. As may be seen, each of these profiles is relatively flat and uniform.

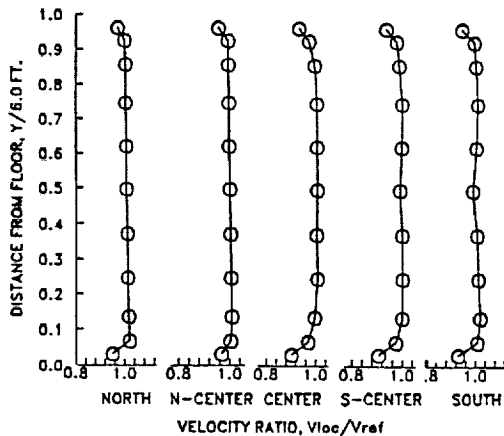


Fig. 3 Measured Profiles Across Diffuser Entrance

Flow in the diffuser exit is markedly different from that measured in the diffuser inlet. Figure 4 shows measured vertical velocity profiles at the centerline of the diffuser exit for test section airspeeds of 150 and 300 mph. As can be seen, these profiles are nonuniform. Figure 5 shows measured vertical velocity profiles at five equally spaced locations across the diffuser exit for a test section airspeed of 200 mph. Not only are these profiles nonuniform, but adjacent profiles are dissimilar. The north and south profiles in Fig. 5 indicate that the flow is tending to migrate toward each of the four corners of the diffuser exit.

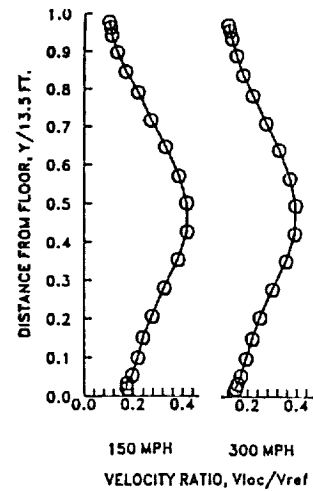


Fig. 4 Measured Profiles at Diffuser Exit

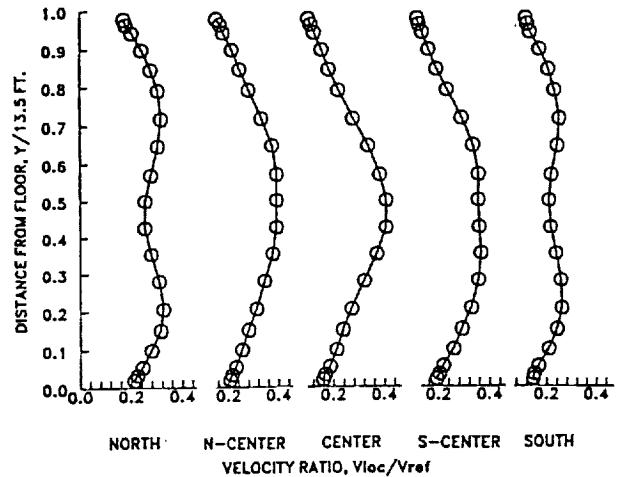
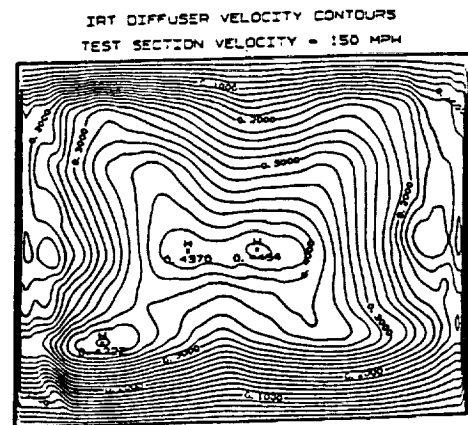


Fig. 5 Measured Profiles Across Diffuser Exit

The contour plot of measured local velocities at the diffuser exit at 150 mph is shown in Fig. 6 and also demonstrates this tendency in the flow.



It is interesting to also note that a similarity exists between the measured airflow velocity contour plot in the diffuser exit and a test section map of peak liquid water content from each spray bar in the system⁵. Figure 7 shows such a map of this peak LWC. This map was established by accreting ice on 2-inch diameter vertical bars mounted in the test section. Each of the eight horizontally mounted spray bars located in the settling chamber of the tunnel were individually operated. The horizontal lines numbered 1-8 in the map indicate the line of maximum ice accretion on each of the vertical bars in the test section. The numbered vertical lines in the map indicate the maximum ice accretion when a single vertical column of nozzles was operated. This similarity indicates the need for further aerodynamic investigation and suggests that these flow characteristics may not be limited to the diffuser.

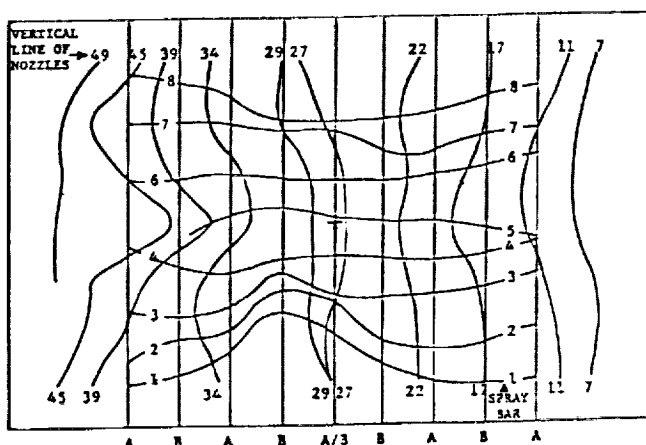


Fig. 7 Map of Peak LWC at Test Section

THE PARC3D COMPUTER CODE

An existing computer code² was used to simulate the flow in the IRT. This code is known as PARC3D. It is based on the complete Navier-Stokes equations written in strong conservation law form. The code uses the Beam and Warming approximate factorization algorithm³ to solve a set of finite difference equations which were produced by central differencing the Navier-Stokes equations on a regular grid. The code calculates flow characteristics based on a specified boundary geometry and the corresponding flow conditions on these boundaries. A wide range of geometries and boundary conditions may be specified.

The flow can be either inviscid or viscous. If viscous, it can be treated as either laminar or turbulent. The turbulence model used is the relatively simple algebraic model of Baldwin and Lomax⁴.

It has been found that the code has had less success when the flow velocity is below a Mach number of 0.1. Moreover, the code does not usually give good results if the grid has high aspect ratio cells. Convergence of the PARC code can be adversely affected by tight grid packing. Use of the turbulence model also tends to slow convergence.

CODE SET-UP FOR IRT SIMULATION

The PARC3D code requires as input: a grid, initial and boundary conditions, and program execution controls. The initial condition, which may also be regarded as an initial solution, does not need to be an accurate one because only steady state results, which are obtained by an asymptotic time march, are of interest. Thus, only a rough, approximate solution is required to get the processing started.

The primary boundary conditions used for the IRT flow simulation runs were: four no-slip, adiabatic walls and subsonic inlet and outlet flows. Uniform distributions of total pressure and temperature were specified on the inflow boundary. At the exit to the diffuser, a uniform static pressure, corresponding to a particular tunnel airspeed, was prescribed. The Reynolds number also had to be specified. For the test section airspeed of 150 mph, the Reynolds number was 1.65×10^6 . The program controls allow the user limited control over program execution such as: limiting the total number of iterations, maximum time step size, tabular output, and so forth.

Several grids were created to reflect the IRT geometry. The initial grid developed for the IRT assumed a one-quarter plane symmetry in the cross section of the tunnel. However, because asymmetric effects had been observed in the tunnel flow, it was decided to use a full cross section model of the tunnel. Such a cross section is shown in Fig. 8. The grid generation scheme developed for this work allowed for grid packing near the walls. This was desired in order to resolve the larger gradients in the flow that were anticipated in these regions. The grids generated had a cross section of 29 points by 29 points and included 55 points in the axial direction. It is recognized that this grid is

a bit coarse, and may not have sufficient grid lines within the wall layers. However, it was deemed an adequate starting structure for this problem.

Two three-dimensional configurations of the tunnel were used. The first was set up to include the aft one-half of the test section and the entire length of the diffuser as shown in Fig. 9. In order to study the effects of the contraction section

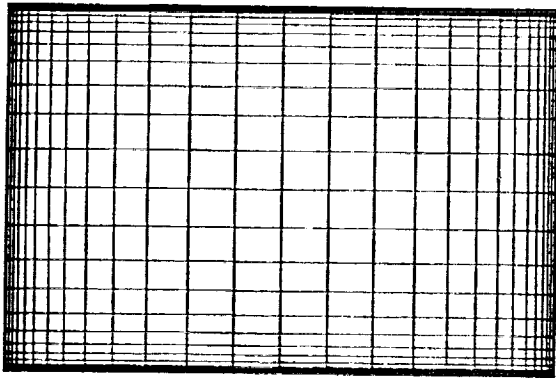


Fig. 8 Full Cross Section for Grid

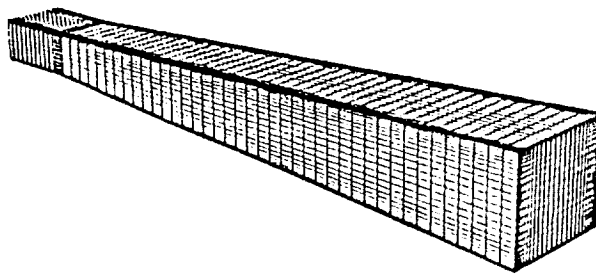


Fig. 9 Mesh for Half Test Section and Diffuser

on diffuser flow, a second configuration was set up which included the settling chamber, the contraction section, the entire length of the test section, and the entire length of the diffuser. This grid is depicted in Fig. 10.

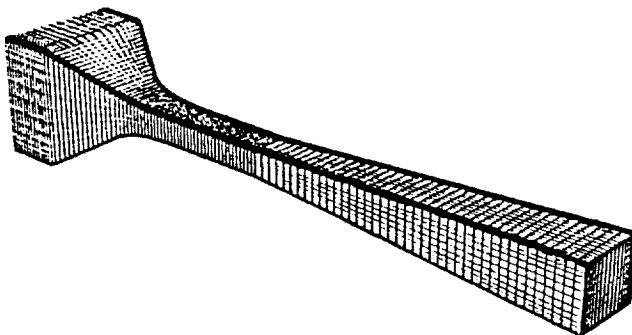


Fig. 10 Mesh for Extended Grid

RESULTS AND DISCUSSION

The results from each particular combination of geometry and flow conditions will be presented through the use of both velocity contour plots and velocity profile plots at key locations along the length of the tunnel. The velocity profile plots provide a more detailed view of the predicted flow characteristics at a particular location while the velocity contour plots give an overall picture of the flow characteristics at the particular cross section of the tunnel.

Even though the main leg of the IRT is geometrically symmetric in both the horizontal and vertical planes, it was felt that the numerical simulations should be run across the entire cross section rather than simply a one-quarter or one-half cross section. In this way, asymmetries due to wall obstacles or flow effects may be subsequently incorporated into the simulation.

The first case was that of turbulent flow for a test section speed of 150 mph. The grid used includes the aft one-half of the test section and the entire diffuser as shown in Fig. 9. The grid dimensions were 29x29x55. Figure 11 shows the velocity contours at the first axial station of the grid which corresponds to a cross section midway down the test section of the tunnel. The contour levels are in terms of Mach numbers with the highest level corresponding to the centermost contour line. As may be seen, the contour lines are bunched relatively tightly together indicating a small boundary layer, and, as might be expected, are very regular and symmetric around the tunnel walls. The boundary layer is slightly thicker at both side walls than at the floor and ceiling.

CONTOUR LEVELS
0.00000
0.02000
0.04000
0.06000
0.08000
0.10000
0.12000
0.14000
0.16000
0.18000

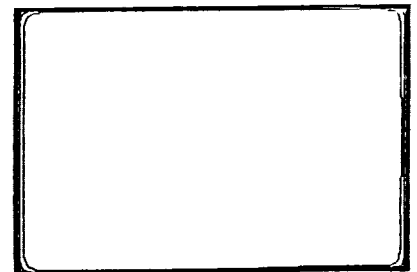


Fig. 11 Predicted Velocity Contours, Test Section

Figure 12 shows predicted velocity contours at the entrance to the diffuser. Again the contours are regular and symmetric around the tunnel walls. The boundary layer has grown slightly toward the center of the tunnel and remains slightly thicker along the side walls. Figure 13 is the predicted vertical velocity profile at the tunnel centerline. The vectors are located at points on the grid and are proportional to the velocity at each point. These profiles also reflect the uniformity of the predicted flow at this location.

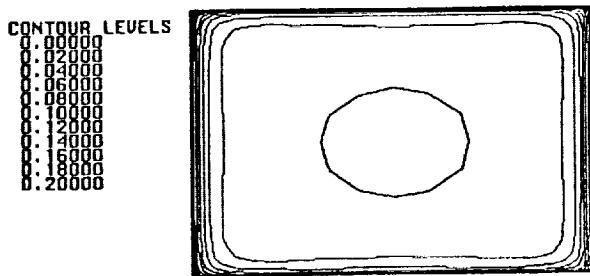


Fig. 12 Predicted Vel. Contours, Diffuser Entrance

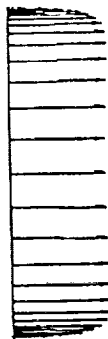


Fig. 13 Predicted Vel. Profile, Diffuser Entrance

Predicted velocity contours at the diffuser exit are shown in Fig. 14. They indicate a flow which is uniformly varying across the tunnel with a small amount of separation in or backflow occurring in each corner. Figure 15 shows the vertical velocity profile at the tunnel centerline. It also depicts the uniformly varying velocity predicted at this location.

To investigate the effects of the contraction section on the flow in the diffuser, the grid was extended to include this part of the tunnel. This includes approximately 10 ft. of settling chamber, the 35 ft. of contraction section and the first 10 feet of the test section. The flow conditions were set to laminar flow for a test section airspeed of 150 mph.

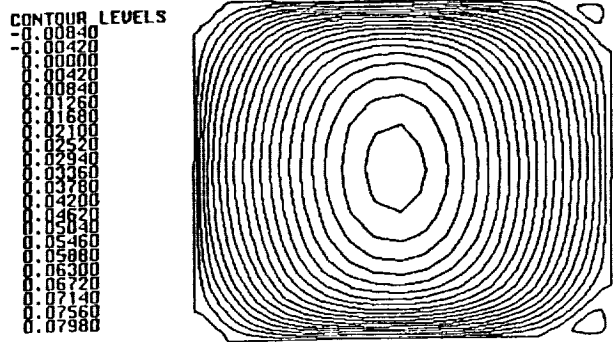


Fig. 14 Predicted Vel. Contours, Diffuser Exit

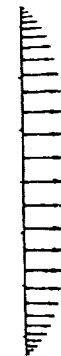


Fig. 15 Predicted Vel. Profile, Diffuser Exit

Figure 16 shows predicted velocity contours for these conditions at a station halfway down the test section. This corresponds to the initial station in the previous case. The contours are regular and symmetric around the walls of the tunnel. The boundary layer is slightly thicker here than in the previous case.

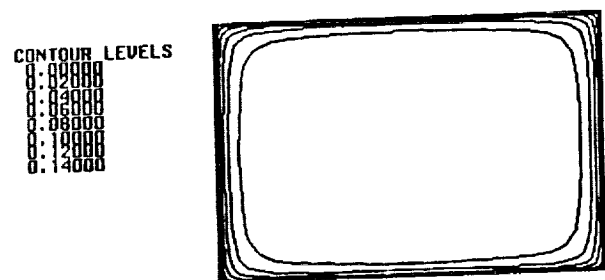


Fig. 16 Predicted Vel. Contour, Test Section, Extended Grid, Laminar

Figure 17 shows predicted velocity contours at the diffuser entrance. Again, the contours are regular and symmetric around the tunnel walls. Figure 18 presents a computed vertical velocity profile at the centerline of the diffuser entrance

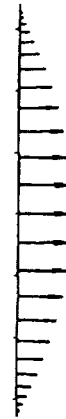
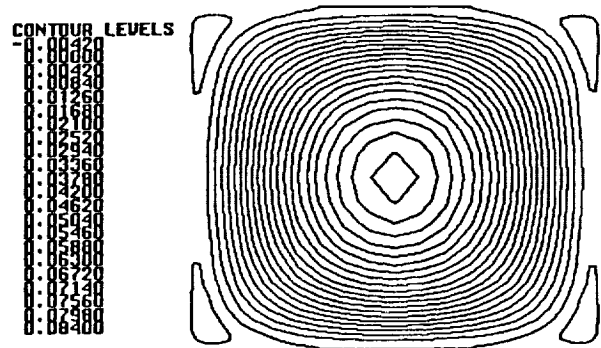
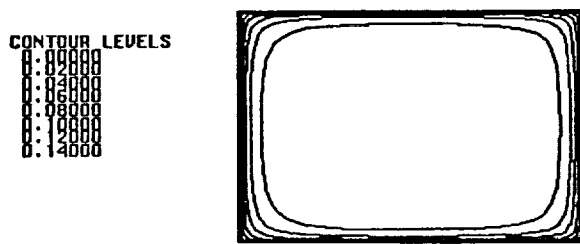


Fig. 17 Predicted Vel. Contours, Diffuser Entrance, Extended Grid, Laminar

Fig. 19 Predicted Vel. Contours, Diffuser Exit, Extended Grid, Laminar

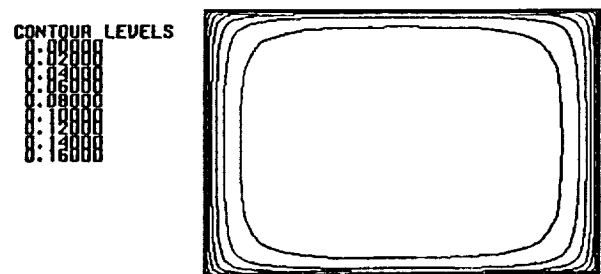
Fig. 18 Predicted Vel. Profile, Diffuser Entrance,
Extended Grid, Laminar

**Fig. 20 Predicted Vel. Profile, Diffuser Exit,
Extended Grid, Laminar**

which also indicates the well behaved flow predicted in this situation.

Predicted velocity contours at the exit of the diffuser are presented in Fig. 19. The flow is well behaved in this case with the velocity uniformly varying across the tunnel. Regions of flow separation or backflow are shown in each of the four corners. Figure 20 displays a predicted vertical velocity profile at the diffuser exit which also demonstrates the uniformly varying velocity across the tunnel here.

The last case to be discussed is similar to the previous case in which the grid includes the contraction geometry, but in this case the turbulence model is activated. Figure 21 shows predicted velocity contours at a station halfway down the test section corresponding to the first station in the first case. As would be anticipated, the boundary layer is slightly thicker than in the first case, but the flow pattern is regular and



**Fig. 21 Predicted Vel. Contours, Test Section,
Extended Grid, Turbulent**

symmetric around the tunnel walls.

Figure 22 presents predicted velocity contours at the diffuser entrance. These contours

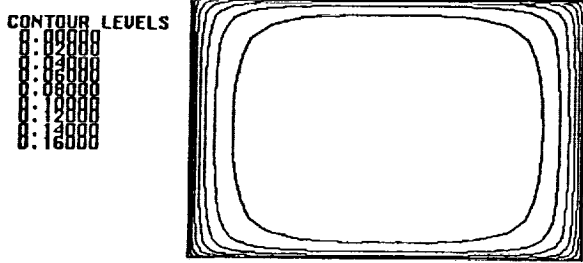


Fig. 22 Predicted Vel. Contour, Diffuser Entrance
Extended Grid, Turbulent

are also regular and symmetric and, in fact, are very similar to the contours at the station halfway down the test section which is 10 feet upstream. A vertical velocity profile is shown in Fig. 23 which also indicates a well behaved predicted flow pattern.

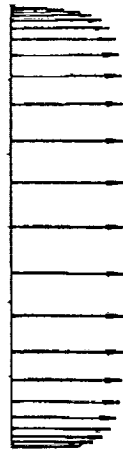


Fig. 23 Predicted Vel. Profile, Diffuser Entrance
Extended Grid, Turbulent

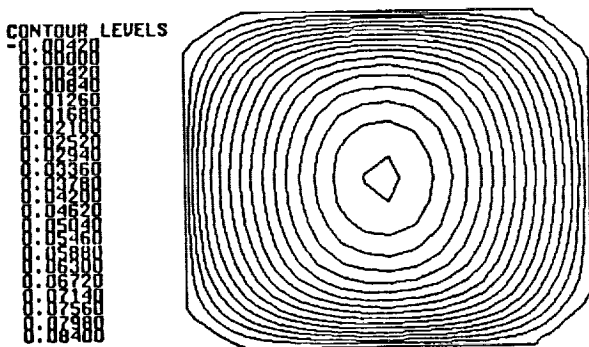


Fig. 24 Predicted Vel. Profile, Diffuser Exit
Extended Grid, Turbulent

Figures 24 and 25 show predicted velocity contours and the vertical velocity profile, respectively, at the diffuser exit. The flow pattern is similar to the previous, laminar flow case, however, less separation or backflow is predicted in each of the four corners of the cross section.

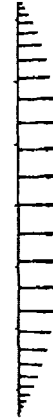


Fig. 25 Predicted Vel. Profile, Diffuser Exit
Extended Grid, Turbulent

In contrasting the measured velocity profiles with those from the computed results, several things are evident. First, the predicted boundary layer thickness for the case with the grid which does not include the contraction section is somewhat thin at the diffuser entrance being about 6% of the tunnel height while both the extended grids agree better, at 9% for the laminar and 11% for the turbulent case. The measured boundary layer thickness is approximately 10% of the tunnel height. Furthermore, in contrasting the shapes of the velocity profiles at the diffuser exit, it is evident that the extended grid, laminar flow case shown in Fig. 20 most closely resembles the shape of the measured profile shown in Fig. 4, than do either of the two turbulent cases shown in Figs. 15 and 25.

CONCLUDING REMARKS

The fundamental purpose of this paper was to numerically simulate the flow in the diffuser section of the IRT. It was found that the PARC3D computer code did a more acceptable job of predicting the flow details than did the previously used PEP SIG code. However, complete agreement with measured velocity values is yet to be realized. Several steps may be taken to improve this agreement:

1. Further refinement of the grid structure is

warranted. In particular, the wall layers require more grid points than were used in the current study.

2. An investigation needs to be performed to ascertain that the predicted results are nongrid dependent.

3. Downstream turning vane effects need to be incorporated into the simulation. These will help to generate the asymmetry conditions seen in the measured flowfield values but not present in the predictions.

4. Flow angularity measurements need to be made in the contraction section and/or the test section to verify that the uniform inflow boundary condition is valid.

Besides the above, it is recommended that the numerical simulation of the IRT include the following:

1. Numerical solution of the energy equation so that tunnel temperature distributions could also be predicted and contrasted to measured values.

2. A particle trajectory code should be employed to predict the pattern of the icing cloud spray.

REFERENCES

¹Addy, Jr., H. E. and Keith, Jr., T. G., "Investigation of the Flow in the Diffuser of the NASA-Lewis Icing Research Tunnel", presented at the 27th Aerospace Sciences Meeting, Reno Nevada, Jan. 1989, AIAA 89-0755

²Cooper, G.K., "The PARC Code: Theory and Usage", USAF paper no. AEDC-TR-87-20, Oct. 1987.

³Beam, R. and Warming, R.F., "An Implicit Finite-Difference Algorithm for Hyperbolic Systems in Conservation-Law Form", J. of Computational Physics, Vol. 22 No. 1 Sept., 1976, pp. 87-110.

⁴Baldwin, B.S. and Lomax, H. "Thin Layer Approximation and Algebraic Model for Separated Turbulent Flows", presented at the 16th Aerospace Sciences Meeting, Huntsville, Alabama, Jan., 1978, AIAA 78-257.

⁵Ide, R., "Liquid Water Content and Droplet Size Calibration of the NASA-Lewis Icing Research Tunnel", presented at the 28th Aerospace Sciences Meeting, Reno, Nevada, Jan., 1990, AIAA 90-0669.

Report Documentation Page

1. Report No. NASA TM-102480 AIAA-90-0488		2. Government Accession No.		3. Recipient's Catalog No.	
4. Title and Subtitle A Numerical Simulation of the Flow in the Diffuser of the NASA Lewis Icing Research Tunnel				5. Report Date	
				6. Performing Organization Code	
7. Author(s) Harold E. Addy, Jr. and Theo G. Keith, Jr.				8. Performing Organization Report No. E-5270	
				10. Work Unit No. 505-68-11	
9. Performing Organization Name and Address National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191				11. Contract or Grant No.	
				13. Type of Report and Period Covered Technical Memorandum	
12. Sponsoring Agency Name and Address National Aeronautics and Space Administration Washington, D.C. 20546-0001				14. Sponsoring Agency Code	
15. Supplementary Notes Prepared for the 28th Aerospace Sciences Meeting sponsored by the American Institute of Aeronautics and Astronautics, Reno, Nevada, January 8-11, 1990. Harold E. Addy, Jr., NASA Lewis Research Center; Theo G. Keith, Jr., University of Toledo, Toledo, Ohio 43202.					
16. Abstract In this paper, the flow in the diffuser section of the Icing Research Tunnel at the NASA Lewis Research Center is numerically investigated. To accomplish this, an existing computer code is utilized. The code, known as PARC3D, is based on the Beam-Warming algorithm applied to the strong conservation law form of the complete Navier-Stokes equations. The first portion of the paper consists of a brief description of the diffuser and its current flow characteristics. A brief discussion of the code work follows. Predicted velocity patterns are then compared with the measured values.					
17. Key Words (Suggested by Author(s)) Diffuser 3-D Icing wind tunnel Numerical simulation			18. Distribution Statement Unclassified-Unlimited Subject Category 09		
19. Security Classif. (of this report) Unclassified		20. Security Classif. (of this page) Unclassified		21. No. of pages 11	
				22. Price* A03	